Pollution Dispersion in Urban Landscapes

By Alan Huber, *National Oceanic and Atmospheric Administration*, *ASMD*, in partnership with *US Environmental Protection Agency*, *National Exposure Research Laboratory*, Research Triangle Park, North Carolina, USA; Matthew Freeman and Richard Spencer, *US EPA Environmental Modeling and Visualization Laboratory*, *Lockheed-Martin Operations Support*, Research Triangle Park, North Carolina, USA; Walter Schwarz, Brian Bell, and Karl Kuehlert, *Fluent Inc.*

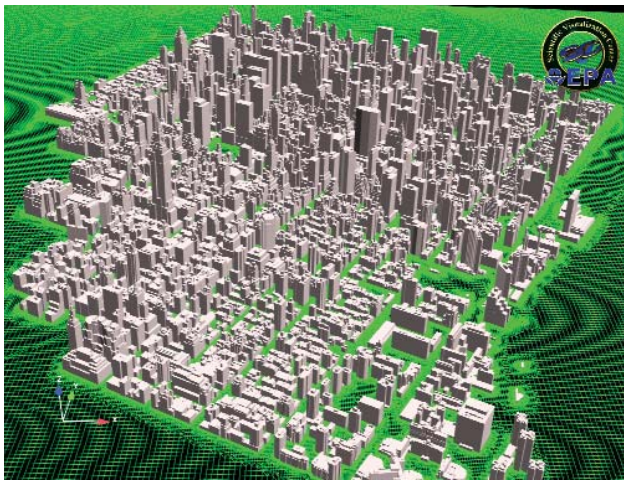
Understanding the pathways of toxic air pollutants from their source through the air humans breathe in urban areas is of critical interest to governmental agencies that have a responsibility to protect the public health and welfare. Rapid assessments of risk, such as the migration of toxic gases related to major fires or chemical spills, are vital to first responders, local officials, federal officials, and the public. The scientific shortcomings are especially serious for incidents that occur in an urban center where the airflow around large buildings is very complex and poorly understood.

High-resolution, high-fidelity CFD simulations have long been used in the aerospace and automotive industries to evaluate the detailed airflow associated with the design and operation of airplanes and cars. CFD techniques also have the potential to be used to describe, for example, the flow of pollutants from accidental explosions, fires, or routine emissions from human activities such as driving motor vehicles; or the release of biological agents associated with an accident or terrorist event.

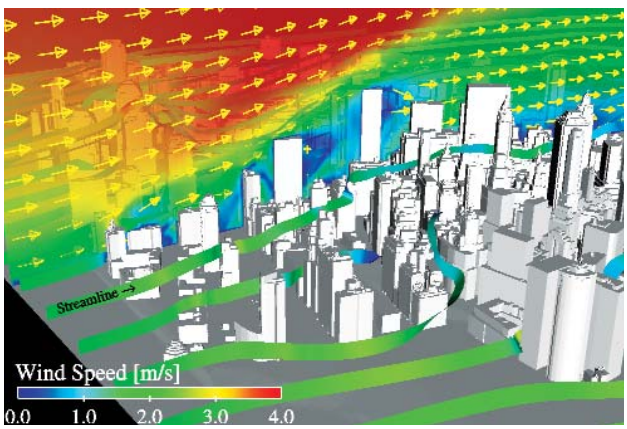
In recent years, CFD modeling has emerged as a promising technology for such assessments. Already in use by early-adopters, CFD has demonstrated the potential to yield accurate solutions because it is based on fundamental physics, on the effects of detailed three-dimensional geometry, and on local environmental conditions. However, today's CFD tools are not well evaluated for environmental modeling. Best-practice methodologies have not been established. Furthermore, today's CFD tools are not tailored to the specific needs and nomenclature of environmental scientists. All of these factors limit the ability of this group of researchers to benefit from CFD as a routine assessment technique.

Through a Cooperative Research and Development Agreement (CRADA), Fluent Inc. and the US Environmental Protection Agency (EPA) have worked towards the evaluation and demonstration of CFD for large-scale environmental applications. Efforts are being made to demonstrate best practices for using CFD as a tool for estimating potential human exposures to local sources of toxic air contaminants in geometrically complex environments.

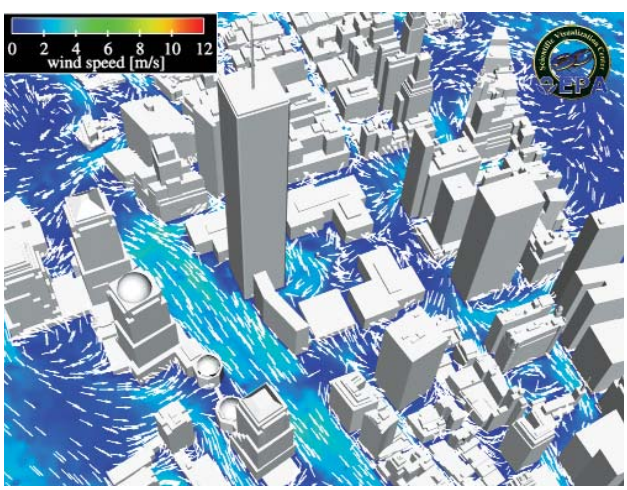
In addition to this general goal,



An example of the detailed building geometry for Manhattan, presented with a surface mesh on the ground; the mesh was created using Harpoon from CEI



A simulation of the complex wind patterns between the buildings of Manhattan, shown using vectors, pathlines, and contours of velocity on a vertical slice



Near surface winds from a CFD solution for lower Manhattan; while the solution has a 1-3m resolution near the ground only about 10% of the vectors are plotted; the winds are from the Northwest (upper left corner)

Fluent has been supporting the US EPA in developing simulations for large sections of Manhattan, which may be one of the most complex building environments worldwide. While it has been a challenging exercise to set up working CFD models that describe the detailed building environment for Manhattan, there have been many lessons learned that are making it easier to set up similarly complex urban environments for future studies. To date the project has focused on steady-state solutions using the widely used k-ε turbulence models. Ongoing developments are being extended to include unsteady solutions and higher order turbulence models as well.

Modeling large numbers of unique buildings typical of major urban areas presents special challenges in developing an applicable computational mesh. The digital geometries of many of the buildings in large cities have already been obtained and are being updated by government agencies and commercial vendors. While the typical building geometries from these sources easily support visual models, minor flaws that may not be visible lead to difficulties for CFD modeling. A great deal of development work has gone into improving and streamlining the clean-up procedures for flawed geometries of buildings and for other objects in general, such as manufactured equipment.

Once the geometry meets certain requirements, it is used for the creation of a surface mesh and subsequently, a volume mesh. For the steady-state simulations of urban areas performed to date, a mesh with computational cells one to two meters in size near the building surfaces has proven sufficient. This results in models with no more than 50 million cells overall, which can be handled by the computing resources available for the project. For future work the present methods are scalable to larger sized models.

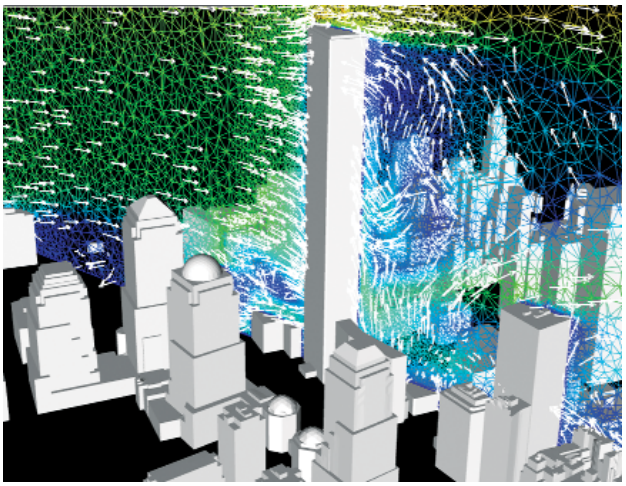
One recent EPA study involved the dispersion of pollutants emanating from Ground Zero following the collapse of the World Trade Center (WTC), and this case has provided a unique opportunity to develop and demonstrate the capabilities of CFD for environmental analysis. Winds in urban micro-environments are very complex because of blockages

and aerodynamic influences from the built environment. Visualizations of the flow on vertical slices through city streets or horizontal slices near ground level illustrate the complexity in the wind patterns. Even if only 10 percent of the solution vectors are plotted in a typical display, the pattern of downward airflow on the windward side of a building and upward flow on the leeward side is clear.

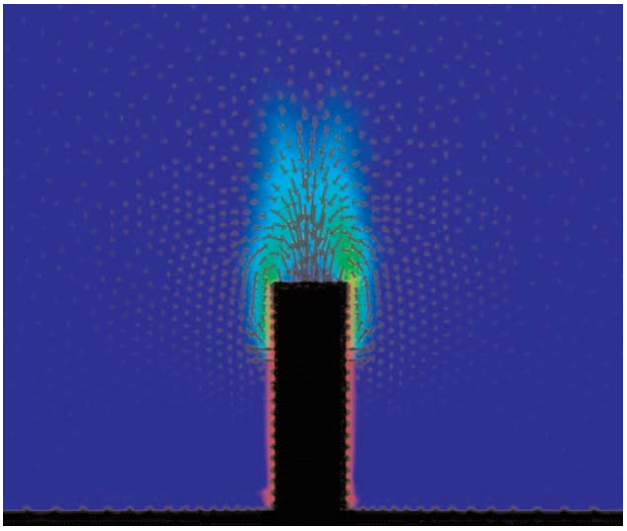
Before the collapse of the WTC, the buildings stored a great deal of potential energy, owing to their extreme height above the ground. During the collapse, this potential energy was converted to kinetic energy, and a large amount of momentum was generated. Each falling building entrained air into the volume previously occupied by the building, forming large circulation currents and strong local winds. In addition to the motion of the tower, the airflow in the vicinity of the WTC during the collapse was governed by the local built landscape and the air and smoke that squeezed out of the building and discharged into the surroundings as individual floors collapsed.

To simulate the fall of the tower, the dynamic mesh model in FLUENT was used. This capability allows the arbitrary motions of walls to be defined. By considering only the deformation of the outer shape of the building, the collapsing tower was essentially modeled as a piston approaching the ground surface. Approximately 90 to 95% of the volume of the WTC was assumed to be air and smoke; the remaining volume contained solid material.

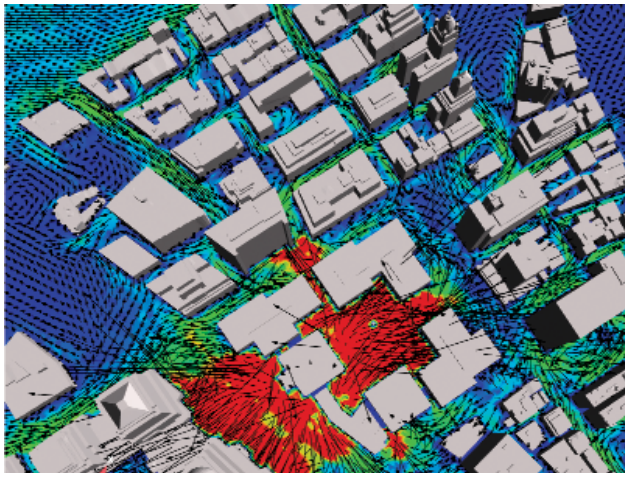
The collapse of the tower takes about 15 seconds. The CFD results indicate that the collapsing tower creates vortex structures in the surrounding air, which transport gaseous constituents and particulate matter radially outward from the base of the buildings. Particles of different size are injected into the airflow from each collapsed floor. The dispersion of the particles is directly controlled by the airflow and turbulence field. The smallest particles remain suspended within the displaced volume of the collapsing building, while the larger particles fall to the ground. The radial impulse created by the collapsed tower, with velocities of up to 30 to 40 m/s, is short- lived however, and soon the material is transported by the prevailing winds through lower



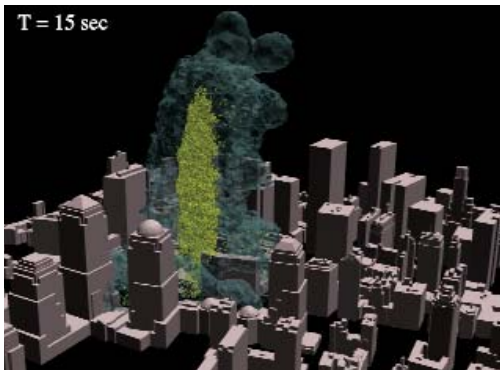
Winds from the northwest and the resulting airflow on a vertical plane in lower Manhattan

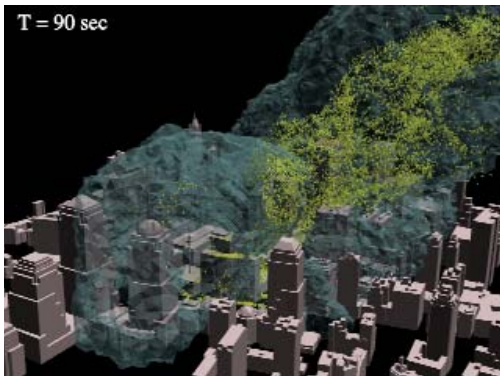


CFD simulation of a collapsing building showing air entrainment into the collapsed volume



Surface winds, 5m above the ground, immediately following the initiation of the building collapse





The outer boundaries of smoke (gray) and a particle cloud (yellow) 15 (top) and 90 (bottom) seconds after the start of the building collapse

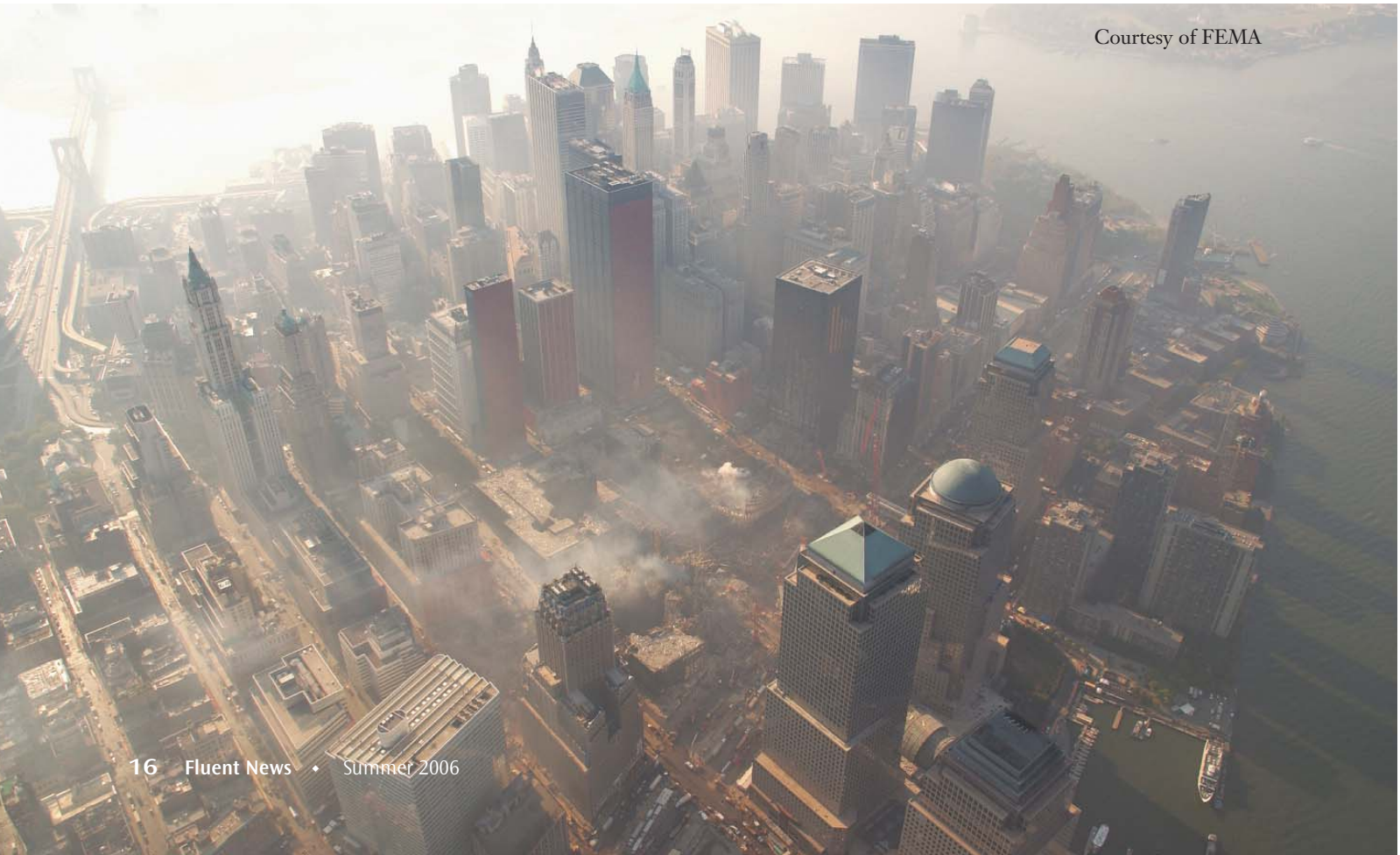
Manhattan and into the surrounding metropolitan area.

The CFD results are being compared to data recorded in an EPA wind tunnel model using post 9/11 conditions with westerly or southwesterly winds. Generally good comparison has been found at several measurement points, particularly in vertical profiles of wind speed. Other variables such as wind direction and turbulent kinetic energy have also been compared. Overall, these results support the CFD methodology.

The application of CFD simulations may evolve for routine environmental use following further demonstrated evaluations with wind tunnel models and field measurements. Measurements alone will rarely be sufficient for accurate planning or understanding what may have happened during an event. Ongoing developments and demonstrations of the reliability and accuracy of CFD simulations will lead to its broad future application for urban air quality and homeland security studies. As has happened for the aerospace and automotive industries, continued evaluation work will make CFD simulations increasingly trusted by environmental engineers for applications such as this. ■

Disclaimer: Although this work has been reviewed by EPA and NOAA and approved for publication, it does not necessarily reflect their policies or views.

Acknowledgement: Scientific visualization and high performance computing were provided through support from the US EPA's National Environmental Computing Center.



Courtesy of FEMA